VII International Conference on Computational Methods in Marine Engineering
MARINE 2017
M. Visonneau, P. Queutey and D. Le Touzé (Eds)

INFLUENCE OF THE DRAFT TO SHIP DYNAMICS IN THE VIRTUAL TANK BASED ON OPENFOAM

P. DU[*], A. OUAHSINE[*], P. SERGENT[†]

[*] Laboratoire Roberval, UMR-CNRS 7337
Sorbonne Universités, Université de Technology de Compiègne
Centre de Recherches Royallieu, CS 60319, 60203 Compiègne cedex, France
e-mail: pp1565156@126.com (P. Du); ouahsine@utc.fr (A. Ouahsine)

[†] CEREMA-134, rue de Beauvais, CS 60039, 60200 Compiègne, France

Key words: Draft, KCS, OpenFOAM, RANS

Abstract. A virtual tank is built based on OpenFOAM. The mesh is created using the 'blockMesh' and the 'snappyHexMesh' utilities successively. The 'interDyMfoam' solver is used to solve the flow fields. The results are compared with the experimental data of the Tokyo 2005 CFD workshop, which show good agreement. The cases with seven different drafts are further simulated. It is found that a small draft can make the ship unstable and weaken the effect of the bulbous bow.

1 INTRODUCTION

For a ship, draft is the vertical distance from the keel to the waterline. In the shipping industry, it is an important criteria for calculating the ship displacement and determining the water depth that a ship can safely navigate. Normally, the ship resistance declines with the decrease of the draft. To achieve a good hydrodynamic performance, a bulbous bow is always designed in front of the ship to modify the water flows around the ship, reduce drag and increase speed, fuel efficiency and stability. However, its effect is only optimized when it is submerged underwater. Besides, a small draft will make the propeller operate near the free surface or even in the air. Therefore the draft of a ship should be controlled properly in real maneuvering. In this paper, a virtual tank is designed based on OpenFOAM [1] and a KCS (KRISO Container Ship) is tested with different drafts. The results are validated using the experimental data of the Tokyo 2005 CFD workshop [2]. The influences of drafts to the ship are finally investigated and concluded.

2 NUMERICAL METHODS

2.1 Governing equations

The incompressible URANS (Unsteady Reynolds-Averaged Navier-Stokes) equations for two-phase flow can be written as [3, 4, 5]:

$$\nabla \cdot \mathbf{U} = 0 \tag{1}$$

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot [\alpha(\mathbf{U} - \mathbf{U}_g)] + \nabla \cdot [\alpha(1-\alpha)\mathbf{U}_r] = 0 \tag{2}$$

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot [\rho(\mathbf{U} - \mathbf{U}_g)\mathbf{U}] = -\nabla p_{rgh} - \mathbf{g} \cdot \mathbf{x} \nabla \rho + \nabla \cdot (\mu_{eff} \nabla \mathbf{U}) + (\nabla \mathbf{U}) \cdot \nabla \mu_{eff} + \mathbf{f}_\sigma \tag{3}$$

where $\mathbf{U}$ is the velocity field, $\mathbf{U}_g$ is the grid velocity considering the mesh motion. $p_{rgh} = p - \rho \mathbf{g} \cdot \mathbf{x}$ is a modified pressure defined in OpenFOAM. $\mathbf{g}$ is the gravitational acceleration. $\mu_{eff} = \rho(\nu + \nu_t)$ is the effective dynamic viscosity, where ν and ν_t are the kinematic and eddy viscosity respectively. ν_t is obtained from a specific turbulence model. In this study, the SST k-ω model is adopted. The pressure velocity coupling is realized using the PIMPLE algorithm.

Equation 2 is the VoF (Volume of Fluid) equation with the artificial compression. $\mathbf{U}_r = \mathbf{U}_l - \mathbf{U}_g$ is the relative velocity between two phases. The subscripts 'l' and 'g' designate '*liquid*' and '*gas*' phases. α is the volume fraction defined as [6, 7, 8]:

$$\begin{cases} \alpha = 0 \quad gas \\ 0 < \alpha < 1 \quad interface \\ \alpha = 1 \quad liquid \end{cases} \tag{4}$$

$\mathbf{f}_\sigma = \sigma \kappa \nabla \alpha$ is the surface tension term, where κ is the mean curvature of the free surface, determined using the equation:

$$\kappa = -\nabla \cdot (\frac{\nabla \alpha}{|\nabla \alpha|}) \tag{5}$$

The physical properties (density, dynamic viscosity) are calculated as weighted averages based on the phase fraction:

$$\begin{aligned} \rho &= \alpha \rho_l + (1-\alpha)\rho_g \\ \mu &= \alpha \mu_l + (1-\alpha)\mu_g \end{aligned} \tag{6}$$

2.2 Mesh generation

The mesh is generated using the 'blockMesh' and 'snappyHexMesh' utilities in OpenFOAM. 'blockMesh' is used to generate the background mesh and 'snappyHexMesh' is used to generate the mesh of the ship. The non-dimensional wall distance y^+ of the boundary layer mesh in this study is in the range of $30 < y^+ < 60$.

As shown in Fig.1, a half domain is created using the '*symmetry*' boundary condition to decrease the mesh number. The region of the ship, especially the bow and the aft is refined to better capture the flow fields. The final mesh has about 1,000,000 cells. The boundary conditions can be observed in Fig.1.

parameters	symbols	full scale	model
Scale factor	γ		57.5
Length between perpendiculars	L_{PP} $[m]$	230	4
Length of waterline	L_{WL} $[m]$	232.5	4.0435
Maximum beam of waterline	B_{WL} $[m]$	32.2	0.5635
Draft	T_0 $[m]$	10.8	0.1878
Displacement volume	∇ $[m^3]$	52,030	0.2737
Block coefficient	C_B	0.6505	0.6505
Longitudinal center of buoyancy	LCB $(\%L_{PP})$	-1.48	-1.48
Wetted surface area without rudder	S_W $[m^2]$	9424	2.8504
	K_{xx}/B	0.4	0.4
Moment of Inertia	K_{yy}/B	0.25	0.25
	K_{zz}/B	0.25	0.25

Table 1: Physical and geometrical parameters of the KCS model

3 COMPUTATIONAL TESTS

The computations are carried out for the KCS, whose parameters are shown in Tab.1. The computational results at $Fr = 0.26$ are compared against the experimental measurements from the Tokyo 2005 CFD workshop [2]. The results show good agreements. However some deviations can be observed behind the ship, demonstrating that the mesh there is still not fine enough.

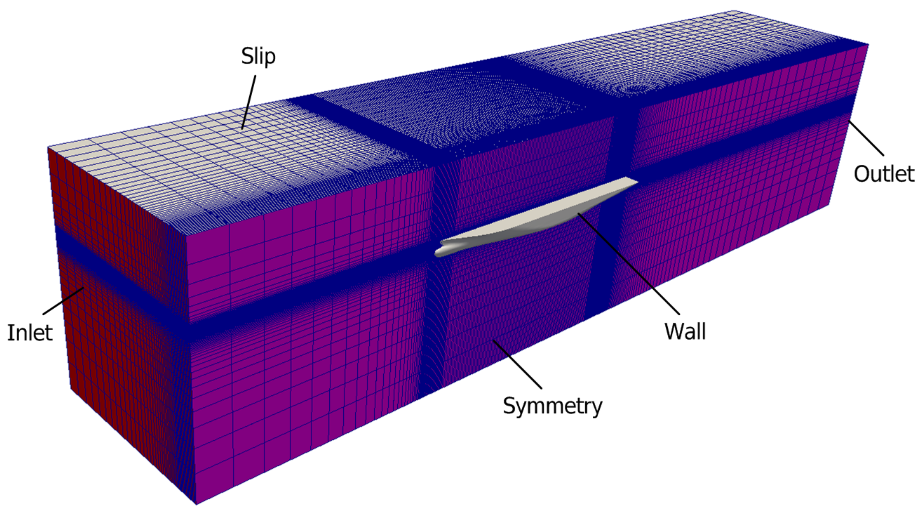

Figure 1: Mesh generation and boundary conditions.

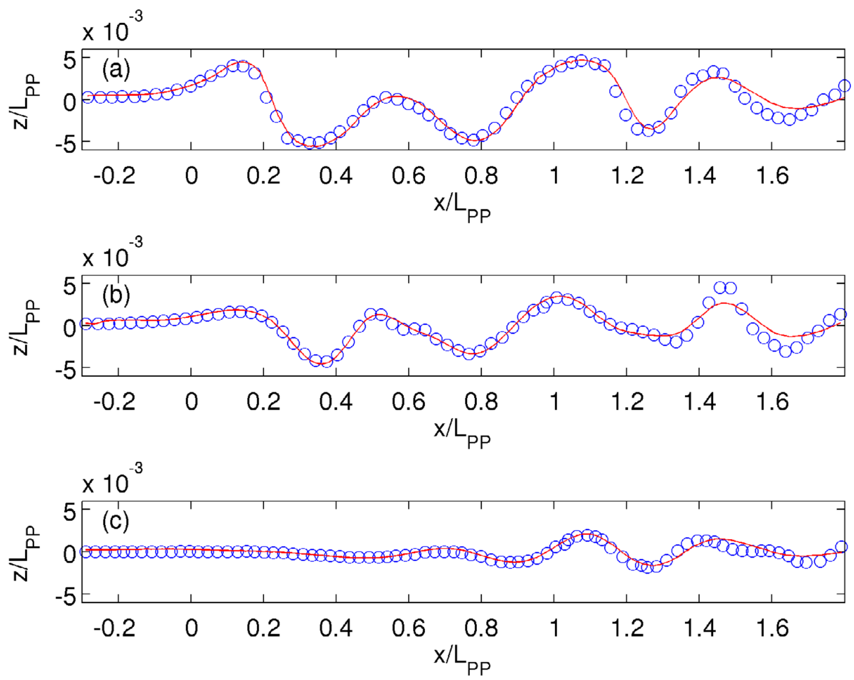

Figure 2: Comparison of wave profiles ($Fr = 0.26$): (a) $y/L_{PP} = 0.0741$; (b) $y/L_{PP} = 0.1509$; (c) $y/L_{PP} = 0.4224$. '$\circ$', experimental results; '-', computational results.

4 RESULTS AND DISCUSSIONS

Simulations with different drafts are conducted to investigate the influence of drafts. The range of drafts in this study is $T = T_0, 0.9T_0, ..., 0.5T_0$. The 'interDyMFoam' solver in the OpenFOAM is used here, which allows the mesh morphing. The free trim and sinkage of the ship can then be realized using this solver. In Fig.5, the mesh morphing can be clearly observed.

In Fig.6, it can be observed that with a smaller draft, the ship takes more time to become stable. Thereby large draft contributes to the ship stability.

In Fig.7(a)-(d), it can be clearly seen that the bulbous bow can modify the flow pattern, which can reduce the drag and increase the ship speed. With the decrease of the draft, the bulbous bow is gradually exposed into the air. Thus its effect of manipulating the flow and reducing the drag will be lost.

5 CONCLUSIONS

- The flow fields of the KCS ship were simulated and showed good agreements with the experimental data.
- Small draft is found to increase the instability of the ship and weaken the effect of the bulbous bow.

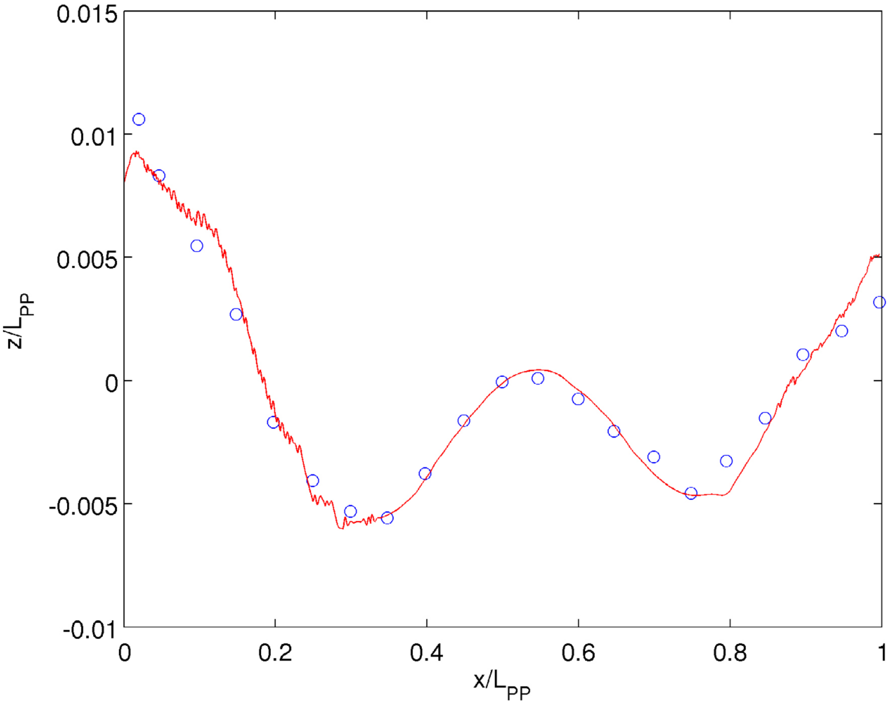

Figure 3: Wave profile on the hull surface ($Fr = 0.26$). '○', experimental results; '-', computational results.

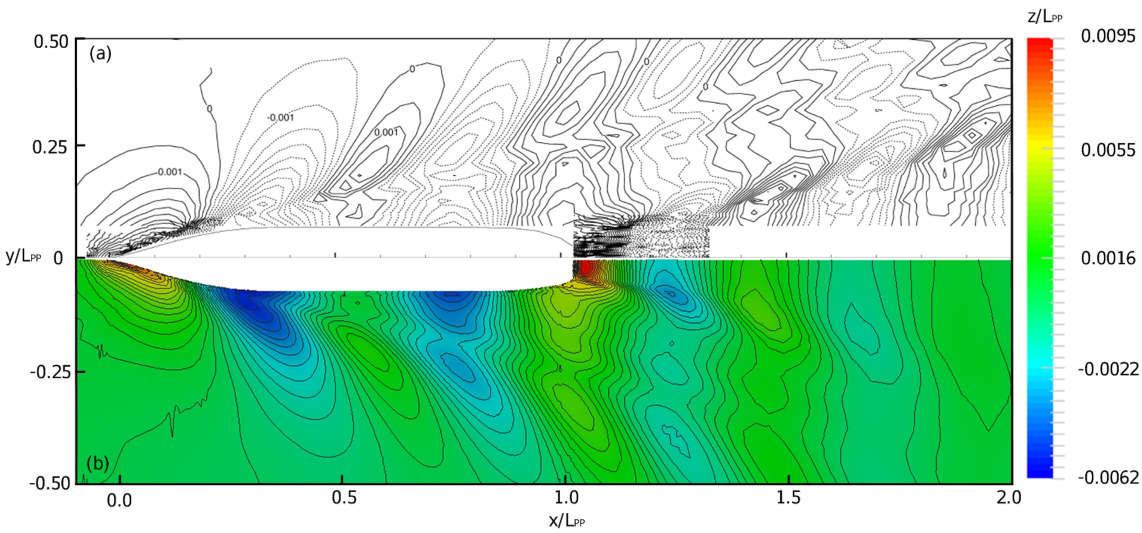

Figure 4: Wave elevation contours ($Fr = 0.26$): (a) experimental results; (b) computational results.

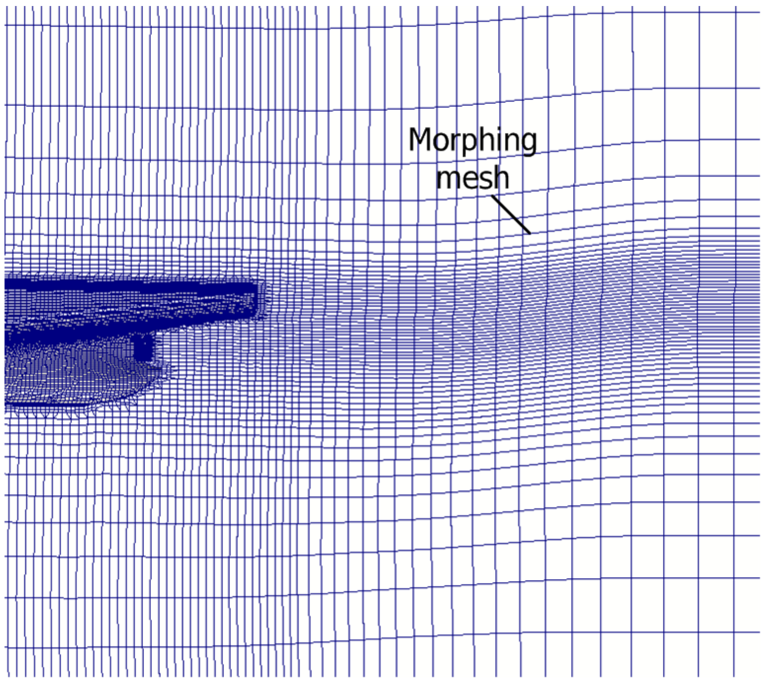

Figure 5: Mesh morphing during the simulations.

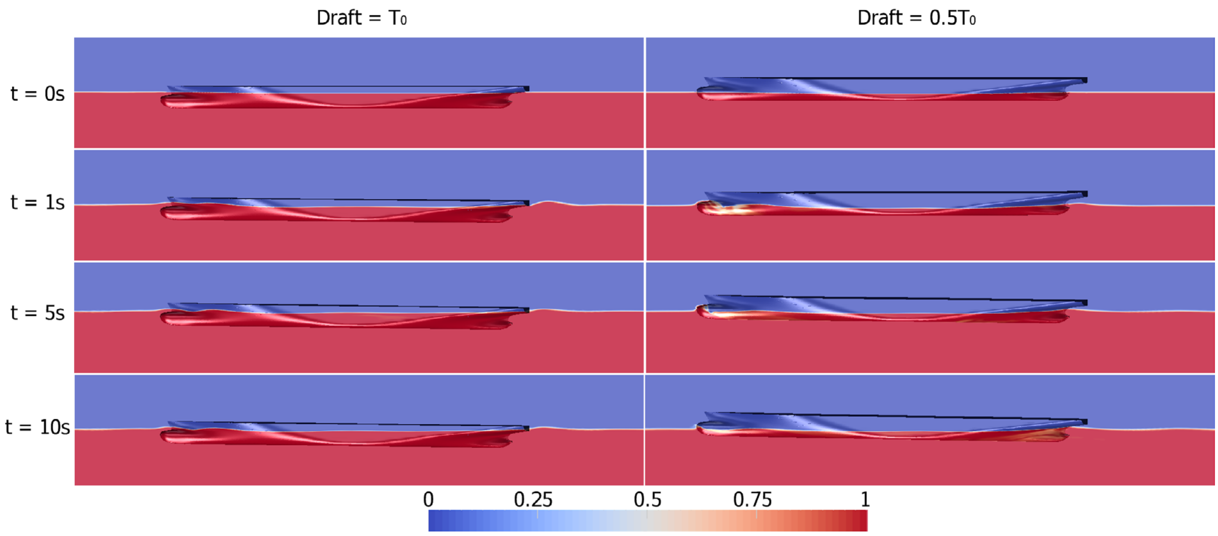

Figure 6: Comparison of unsteady simulations with different drafts (Fr = 0.227). T_0 is the original draft in Tab.1.

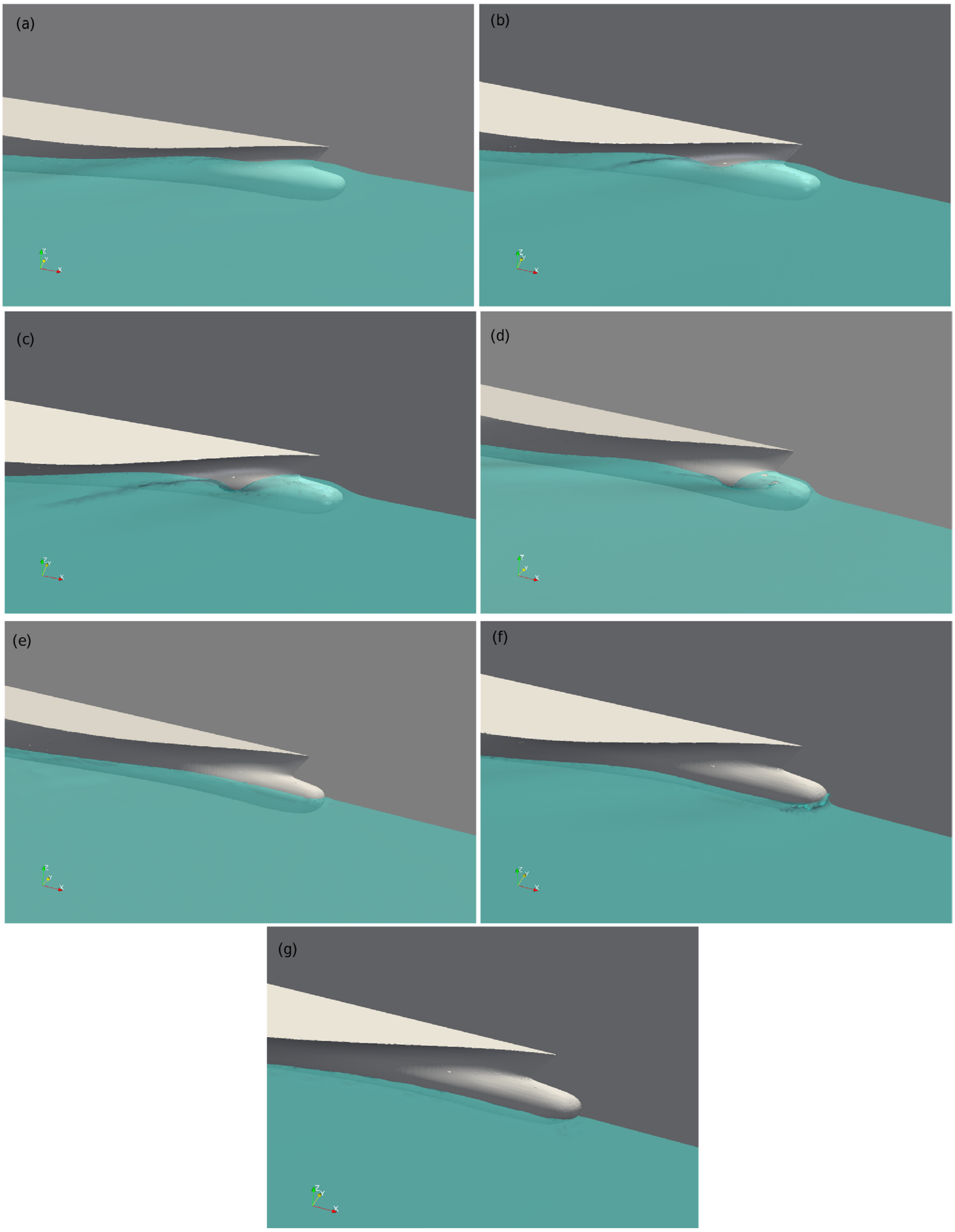

Figure 7: Free surfaces with different drafts ($Fr = 0.227$): (a) $T = T_0$; (b) $T = 0.9T_0$; (c) $T = 0.8T_0$; (d) $T = 0.7T_0$; (e) $T = 0.6T_0$; (f) $T = 0.5T_0$; (g) $T = 0.4T_0$.

REFERENCES

[1] H. Jasak, *Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows, 1996.* PhD thesis, Ph. D. Thesis, University of London Imperial College, 1996.

[2] T. Hino, "Proceedings of cfd workshop tokyo 2005," *Tokyo, Japan*, 2005.

[3] Z. Shen, D. Wan, and P. M. Carrica, "Dynamic overset grids in openfoam with application to kcs self-propulsion and maneuvering," *Ocean Engineering*, vol. 108, pp. 287–306, 2015.

[4] S. C. Ji, A. Ouahsine, H. Smaoui, and P. Sergent, "3d numerical modeling of sediment resuspension induced by the compounding effects of ship-generated waves and the ship propeller," *Journal of Engineering Mechanics*, vol. 140, no. 6, p. 04014034, 2013.

[5] F. Linde, A. Ouahsine, N. Huybrechts, and P. Sergent, "Three-dimensional numerical simulation of ship resistance in restricted waterways: Effect of ship sinkage and channel restriction," *Journal of Waterway, Port, Coastal, and Ocean Engineering*, vol. 143, no. 1, p. 06016003, 2016.

[6] S.-C. Ji, A. Ouahsine, H. Smaoui, P. Sergent, and G.-q. Jing, "Impacts of ship movement on the sediment transport in shipping channel," *Journal of Hydrodynamics, Ser. B*, vol. 26, no. 5, pp. 706–714, 2014.

[7] S.-C. Ji, A. Ouahsine, H. Smaoui, and P. Sergent, "3d modeling of sediment movement by ships-generated wakes in confined shipping channel," *International Journal of Sediment Research*, vol. 29, no. 1, pp. 49–58, 2014.

[8] S. C. Ji, A. Ouahsine, H. Smaoui, and P. Sergent, "3-d numerical simulation of convoy-generated waves in a restricted waterway," *Journal of Hydrodynamics*, vol. 24, no. 3, pp. 420–429, 2012.